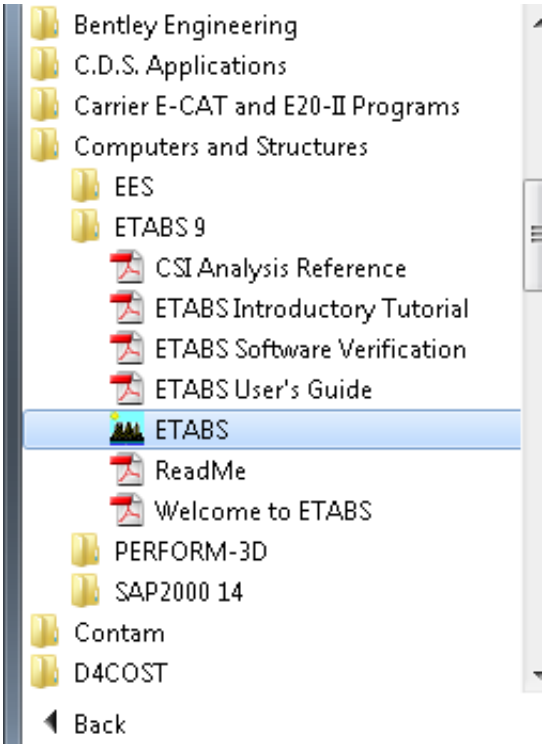
Quick Typical Bay Design in ETABS

-This guide will show you how to get a ballpark estimate of beam and column sizes for a typical bay in your structure

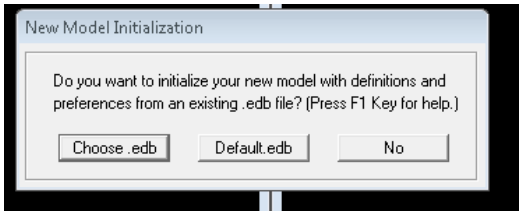
1. Open ETABS

Go to All Programs---Computers and Structures Folder---ETABS Folder---ETABS

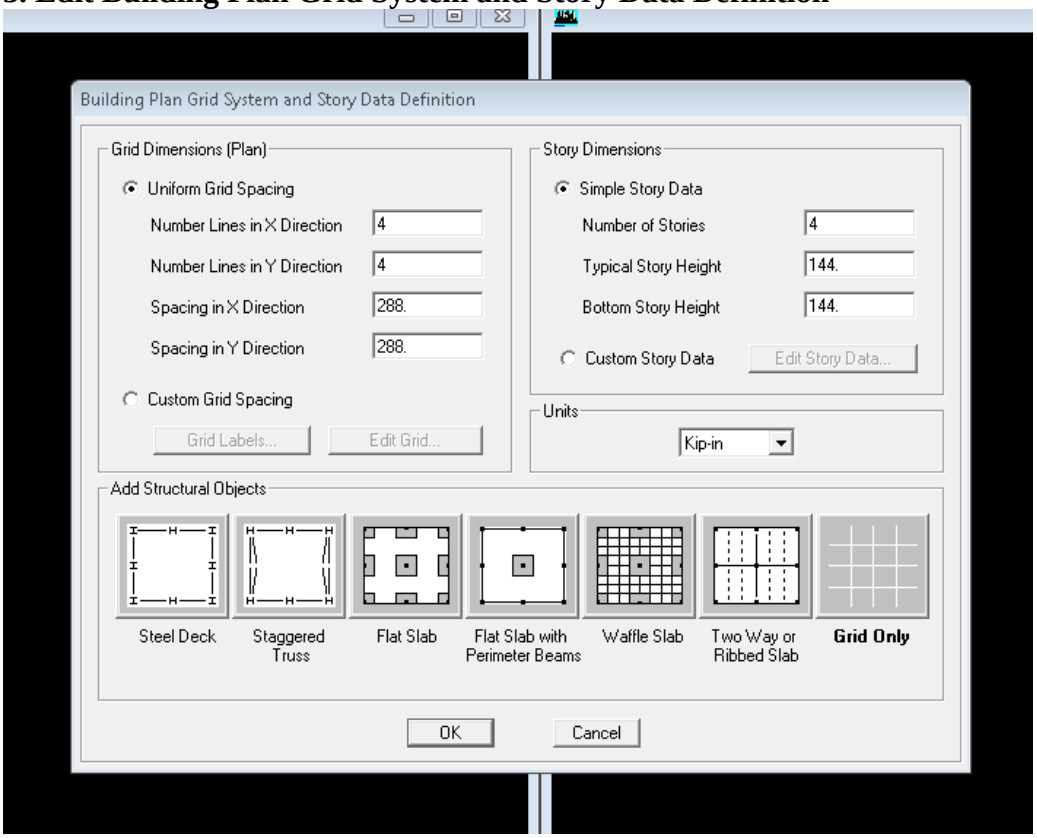


2. Set up New Model

Select File---New Model---Answer No to the question asked

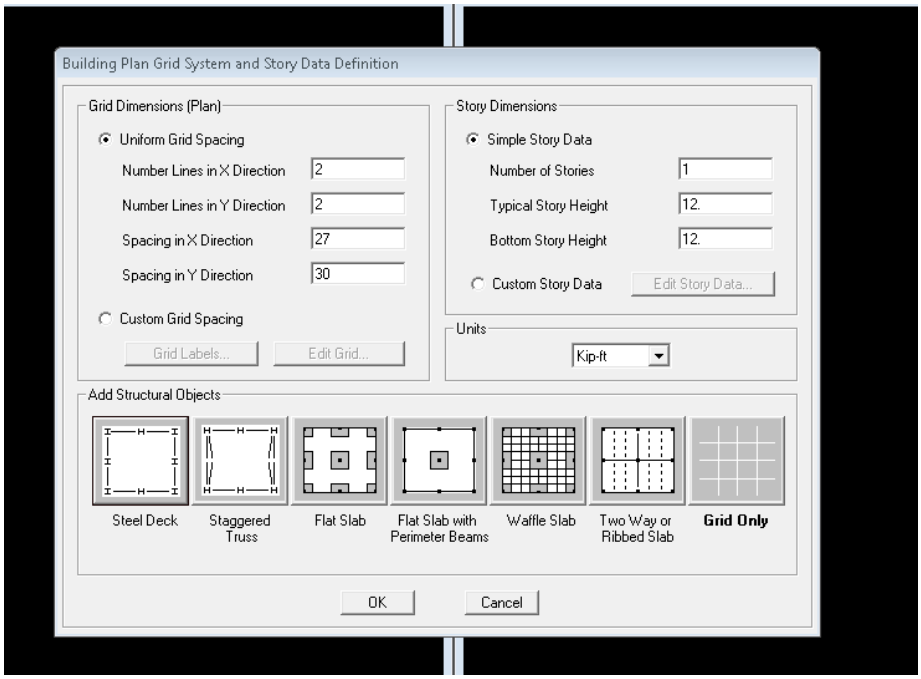


3. Edit Building Plan Grid System and Story Data Definition



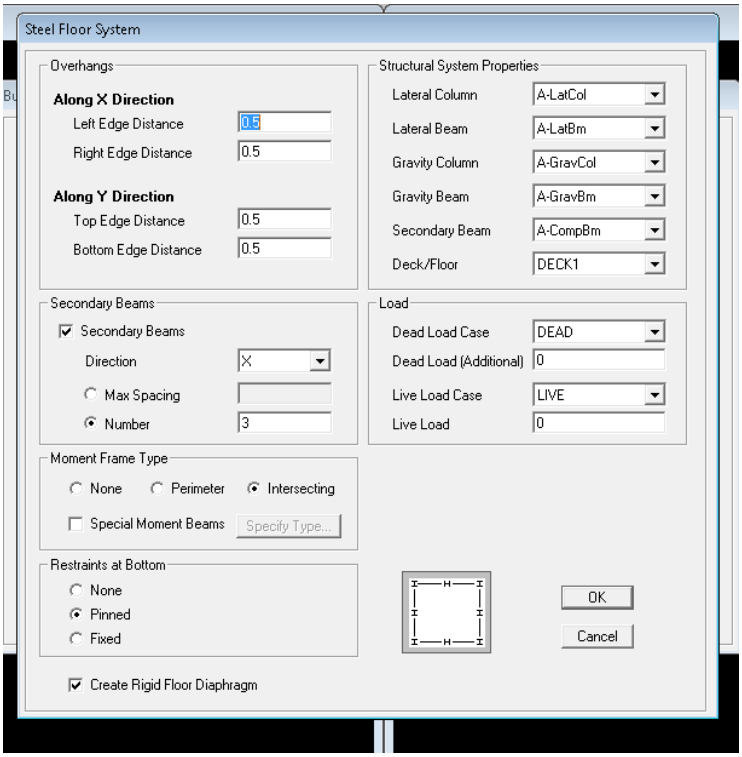
Note the units selected.

For this example, I will just be doing a one story bay. Input as follows:

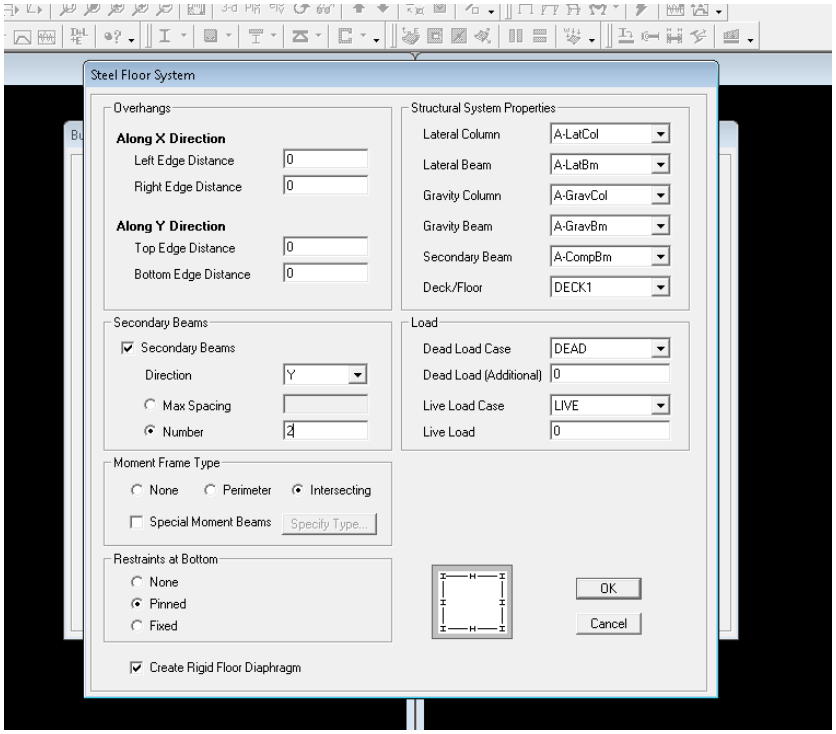


4. You can then select the Structural System that you are designing. This example will show you how to do a Steel System with Steel Deck.

After editing the dimension values, click steel deck.

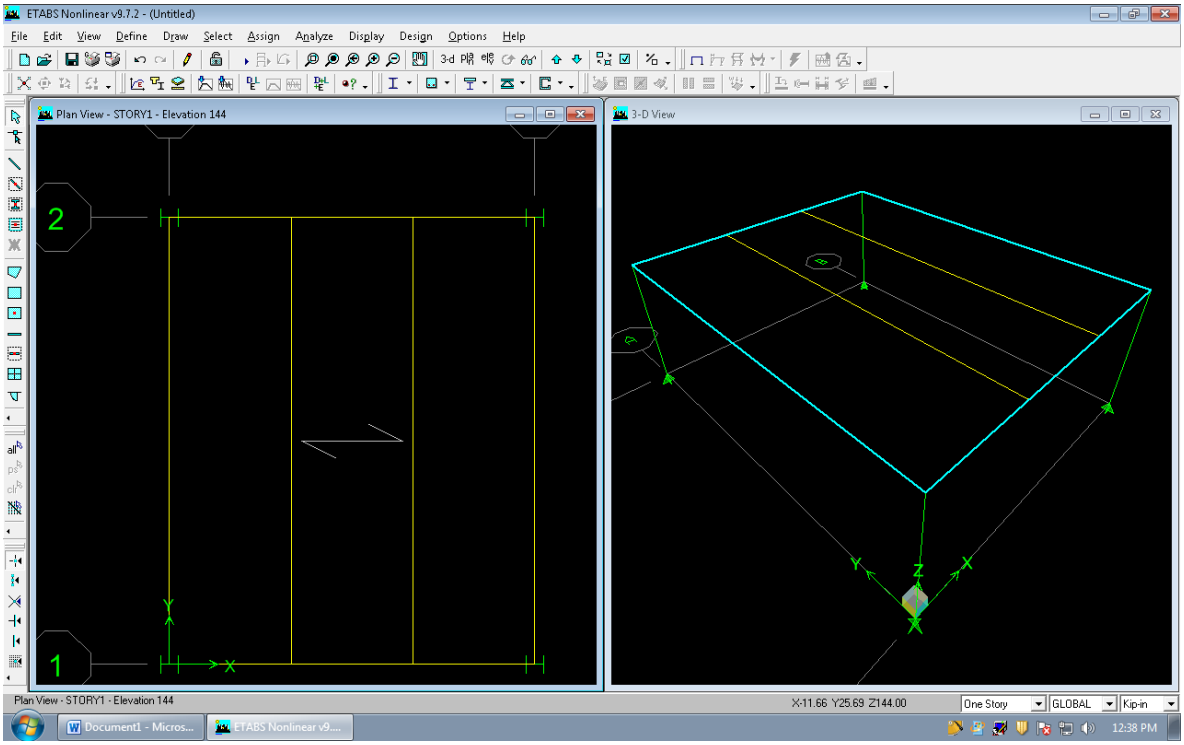


You can now edit the floor system to your typical bay.



Note: You can specify Superimposed Dead Load and Live Load in this step or later. I chose to specify it later.

Then Click Ok and then Ok again and you're bay should appear on screen similar to below.

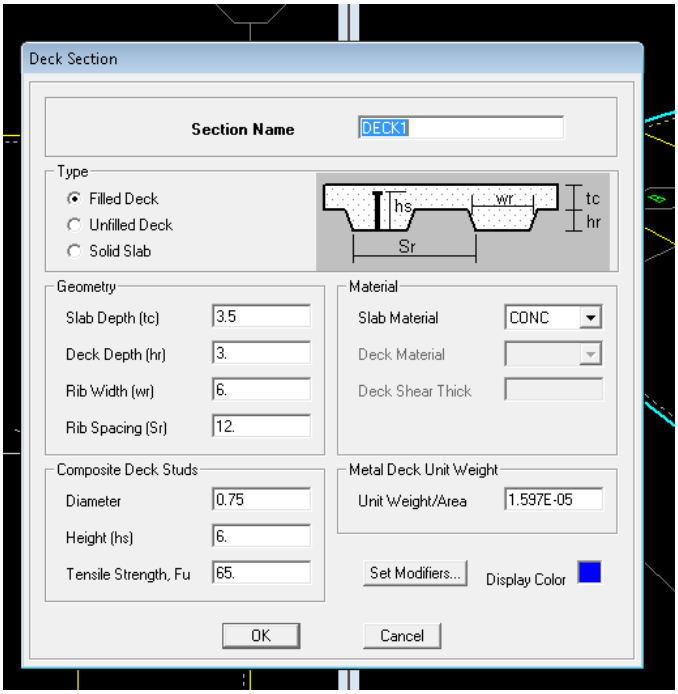


5. Edit your Deck Dimensions

Go to Select---by Area Object Type---Floor---Click Ok
Your floor deck should be selected

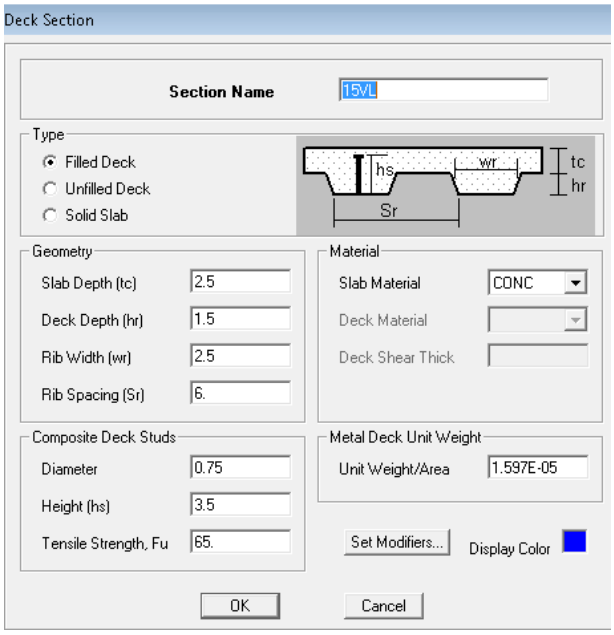
Next, Go to Assign---Shell/Area---Wall/Slab/Deck Section---Choose Deck 1---Modify Section

The window below should appear.



Edit the information to correspond with the floor deck that you have designed.

For example,



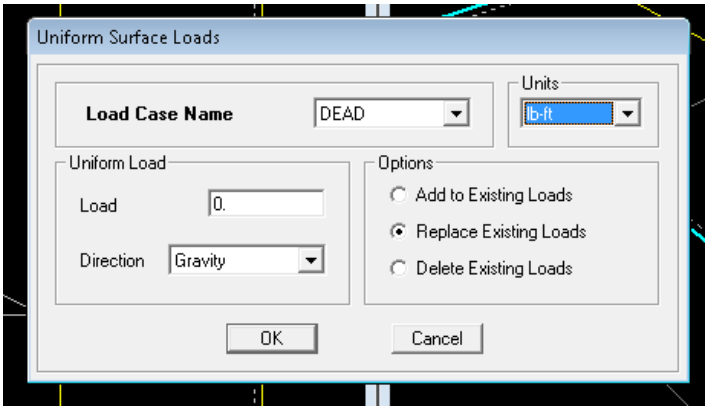
6. Place Loads on the Structure

Go to Select---by Area Object Type---Floor---Click Ok

Next, Go to Assign---Shell/Area Loads---Uniform

Here you can add Superimposed Dead Loads as well as Live Loads

Note the units you select here so that you input the correct load (typically loads are in psf)



Assign as many loads as you want to apply following the same procedure

Note the Load Case you want to place the load in and whether you want to add to existing loads, replace existing loads, or delete the existing loads in that load case.

You may also add lateral loads, but for this example I will only consider gravity loads

7. Run Analysis

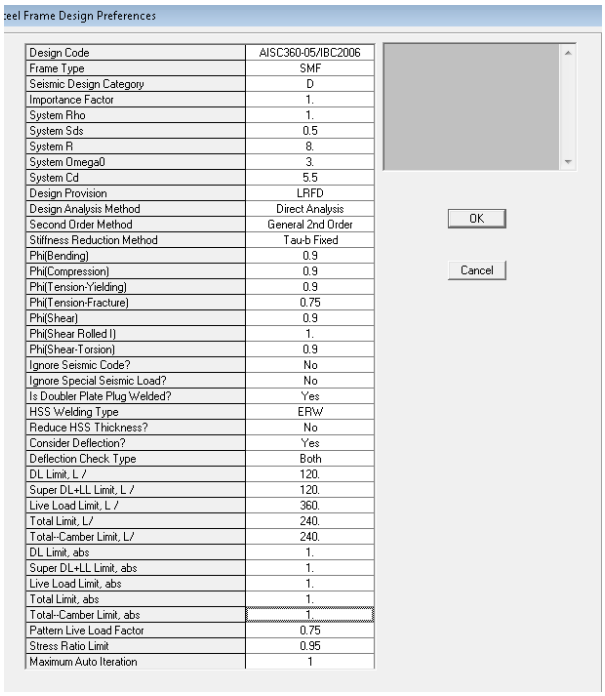
Go to Analyze---Run Analysis

Now, we are ready for the design procedure.

8. Set Design Options

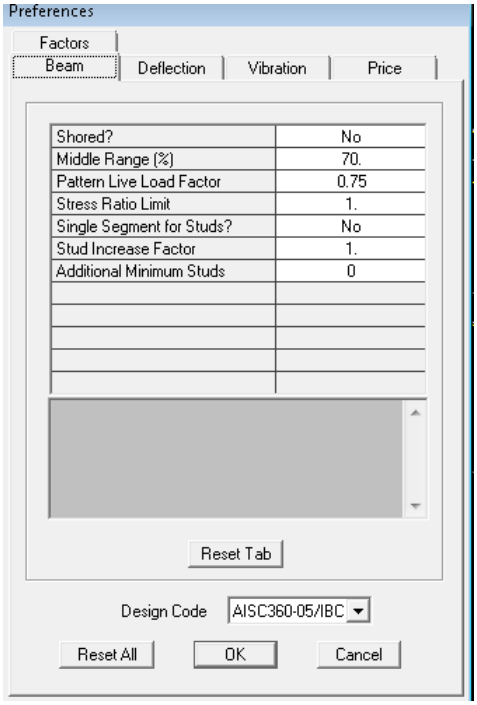
Go to Options---Preferences---Steel Frame Design

Now you can set design parameters and the design code you would like to use.



Design Code	AISC360-05/IBC2006
Frame Type	SMF
Seismic Design Category	D
Importance Factor	1.
System Rho	1.
System Sds	0.5
System R	8.
System Omega0	3.
System Cd	5.5
Design Provision	LRFD
Design Analysis Method	Direct Analysis
Second Order Method	General 2nd Order
Stiffness Reduction Method	Tau-b Fixed
Phi(Bending)	0.9
Phi(Compression)	0.9
Phi(Tension-Yielding)	0.9
Phi(Tension-Fracture)	0.75
Phi(Shear)	0.9
Phi(Shear Rolled I)	1.
Phi(Shear-Torsion)	0.9
Ignore Seismic Code?	No
Ignore Special Seismic Load?	No
Is Doubler Plate Plug Welded?	Yes
HSS Welding Type	ERW
Reduce HSS Thickness?	No
Consider Deflection?	Yes
Deflection Check Type	Both
DL Limit, L /	120.
Super DL+LL Limit, L /	120.
Live Load Limit, L /	360.
Total Limit, L/	240.
Total-Camber Limit, L/	240.
DL Limit, abs	1.
Super DL+LL Limit, abs	1.
Live Load Limit, abs	1.
Total Limit, abs	1.
Total-Camber Limit, abs	1.
Pattern Live Load Factor	0.75
Stress Ratio Limit	0.95
Maximum Auto Iteration	1

Also set the design options for a composite steel design following the same procedure.



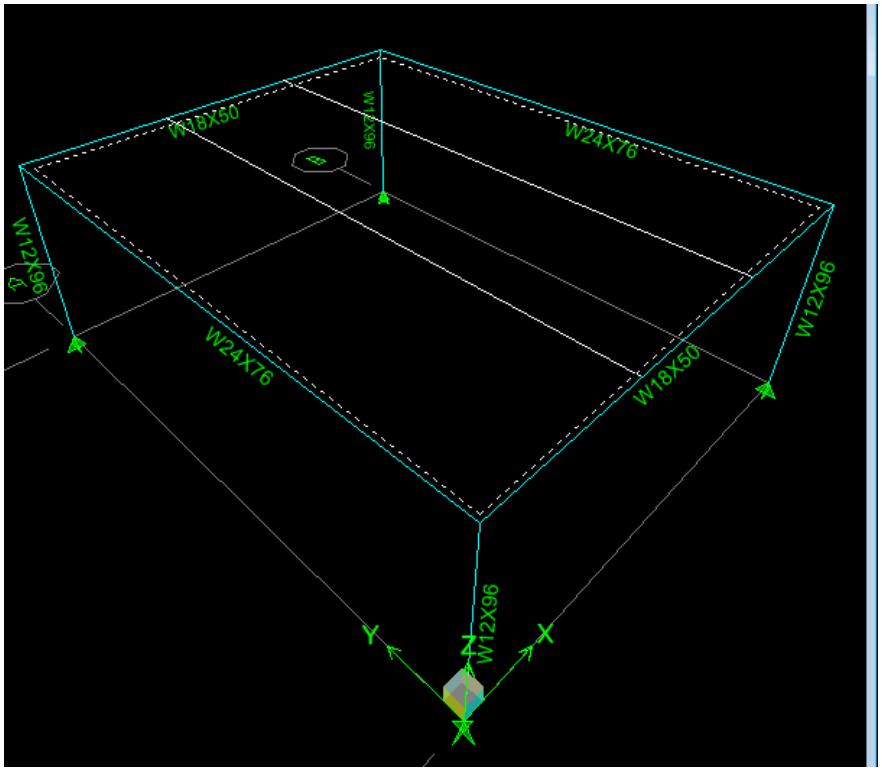
Note you can remove camber options in the deflection tab

By doing this, you can compare design results accurately between the two systems.

9. Design Structure

Go to Design---Steel Frame Design---Start Design/Check of System

The sizes will now appear on the structure like below



You can then follow the same procedure to do a composite steel design and compare your results

Conclusion

You now have a basic range of sizes to expect when designing your structure
It is recommended that you do calculations by hand or by using Excel, to ensure your design satisfies code requirements.